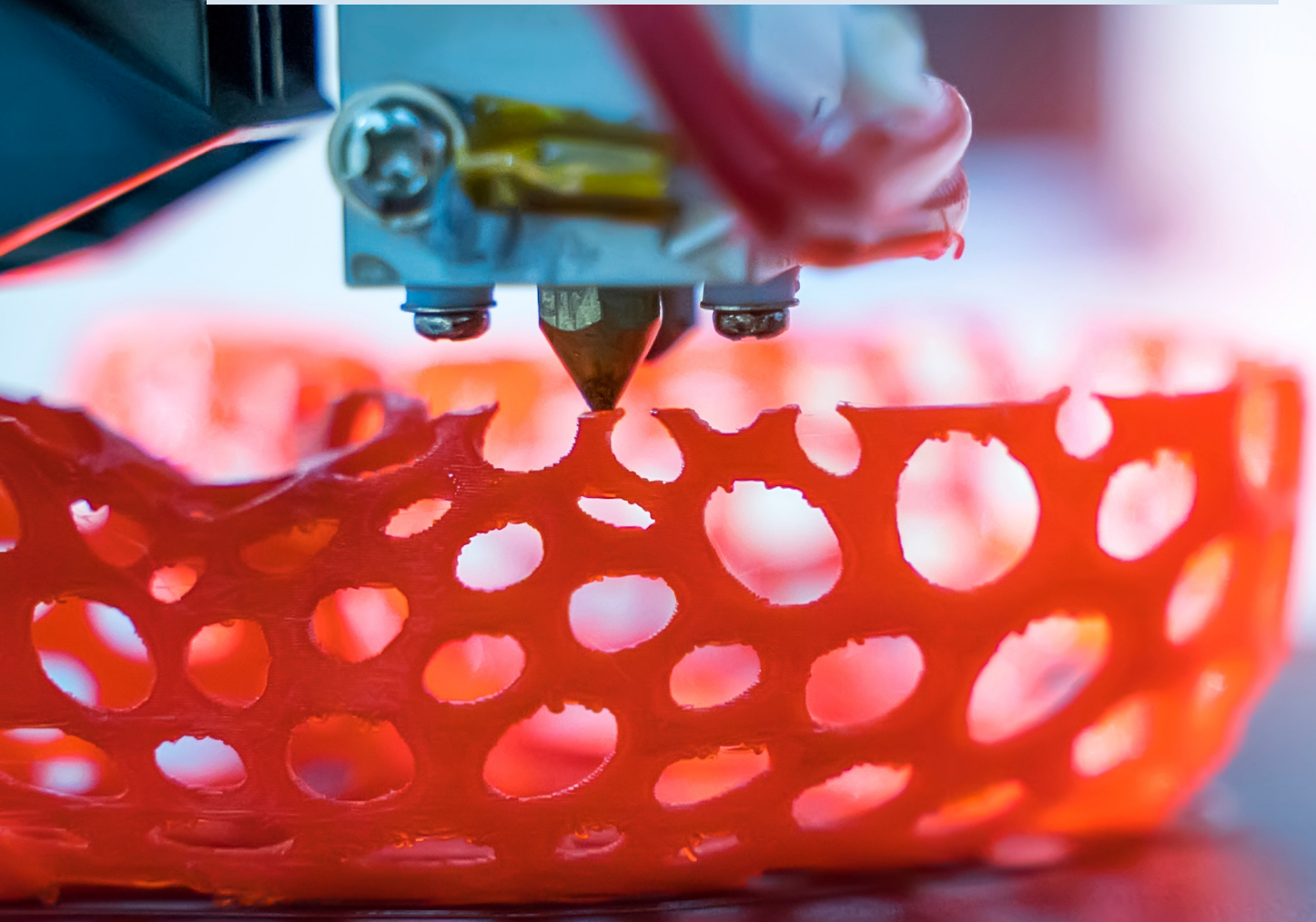
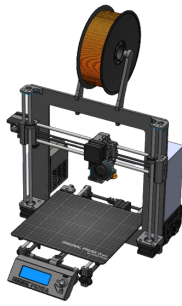


INTRODUCTION TO **3D Printing**

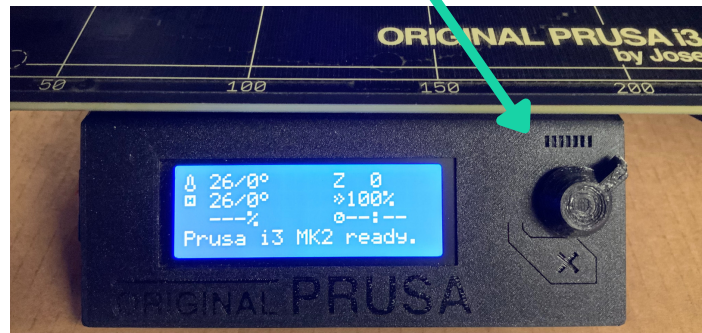
USING THE WLPL PRUSA



The machine you'll be using!

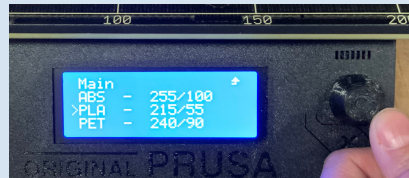


Meet the **Prusa**! It is a Prusa Original i3 mk2 - so an older machine, but pretty reliable! It communicates with you via its **LED screen** and you communicate with it by using the little **dial** to scroll and click the menu options.

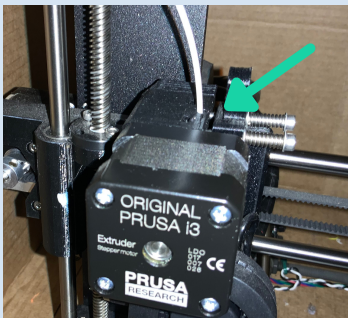


Changing the filament

To change the filament, you'll want to **preheat** the hotend of the extruder by clicking the dial once, then scrolling down to "**Preheat**" then choosing "**PLA**" and clicking again.



The readout shows you the temperatures of the hotend and of the bed (the printing surface). When it reaches its goal temperature (so the readout says 215/215 for example) you can click the dial, then scroll to choose "**Unload**". The motor in the extruder will throw it into reverse pretty quickly, so you have to be ready to pull the filament out of the top.



The extruder will stay hot for a little while, but if you don't have your next color ready to go, you might have to preheat again, so we suggest getting the spool situated in the case before beginning the process of changing it. After pulling out the last filament, click the dial and this time choose "**Load**". The motor will start, so all you need to do is feed the filament down through the little hole in the top of the extruder until you feel it catch.

The Prusa will prompt you a couple times to ask if the filament color is feeding clear out of the extruder. Usually it takes a couple times of choosing "No" to get all of the previous color out of the hotend.

Once you see the color extruding clearly, you're ready to print!

The file you'll need!

The most common file type and the easiest to work with in the **PrusaSlicer** software is an **STL** file. You might also see **OBJ** files, which will also work, but might need some fixing in the slicer software. What you **DON'T** want to be working with is anything in a ready-to-print format like **.gx** or **.gcode**. These types of file are the kind you will be sending to the printer when you are all done tinkering with them - they cannot be changed (or at least not easily!)



There are a few ways to get an STL file - the first and easiest way is to **download** one off an open source website like **Thingiverse.com**. This is a place where a 3D designer can upload their creations for others to download for free. Usually this comes with the typical agreement that no one will sell the design as their own, etc. For a casual user who just wants something fun to print, this is the perfect place to go hunting for designs.

One thing to note is that it's important to read the **design description** and comments - just because a design has been uploaded to Thingiverse doesn't always mean it'll print. See if the creator has comments about whether or not they printed it with supports or what settings they used. You can also check the **comments** and **"makes"** sections to see if others ran into trouble and if they had to use any fixes while printing. Someone might have even made a **"remix"** of the original design, fixing an issue the creator didn't catch. Sometimes it really takes a village!

Another way to get an STL file is to create one yourself! There are a lot of good **CAD** programs out there for 3D design, but one of the easiest and cheapest (it's free!) is **TinkerCad**. It is actually a browser app, so you don't need to download anything and it has a good **tutorial** built in when you start. It walks you through design, customization and ultimately exporting your design as an STL file that you can easily 3D print!



Once you have an STL file in hand, you are ready to "slice" it in **PrusaSlicer**. Slicing a 3D file just means you are converting it to the many little layers the 3D printer needs to create to build the object. In order to do *that*, you need to tell it how thick you want the layers (that's the **resolution**) and how solid you want the interior of the object (that's called **infill**).



It's worth it to note, if you have a hollow object you want to print like a vase, the design will already be hollow, so you don't have to worry about setting infill to 0%. This is only for objects that have no visible openings and will affect how heavy, sturdy or translucent the object is.

So let's have a more in-depth look at PrusaSlicer...

Finally, the software!



Find the STL file you would like to work on then drag the icon on top of the PrusaSlicer icon (shown above). This is the simplest way to open the slicer with your file already loaded.

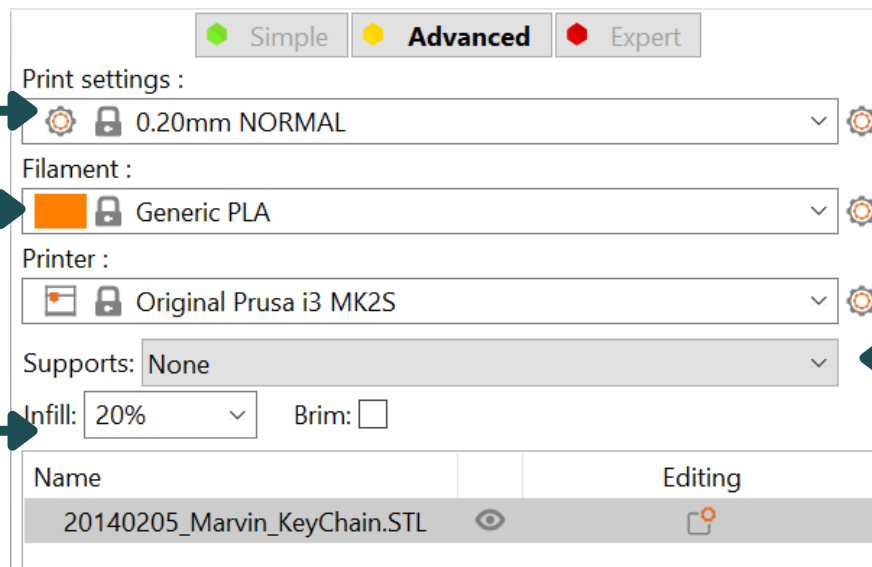
The first thing you'll notice when you open PrusaSlicer is that it can look a BIT complex. The workspace has three settings for "Beginner" "Advanced" and "Expert", which differ in the options that are available to change. For most prints, "Beginner" is just fine, but we'll work in the default "Advanced" setting - we fear no slicer!

Start by checking this section in the top right:

Change the resolution here. 0.2 mm is fine for most prints, but try 0.15 or 0.1mm on a small print to see the difference!

This setting tells the printer what filament you are using so it heats up appropriately.

Tell the printer how much infill - or how solid you want your print. 20% is a good default. For items like dice or keychains, you might want a bit more!



To print with supports (such as when your print has overhangs greater than a 45 degree angle) we'll actually use a different setting, so you don't have to touch this one.

Now on the left side:



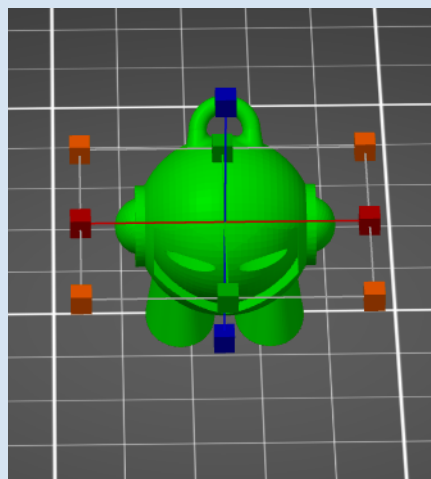
Select this tool to move your object around the plate.

Select this tool to scale your object freehand (there's another spot to scale more precisely)

Select this tool to rotate your object freehand.

This tool rotates a specific surface of the print to the plate..

Let's not worry about this stuff right now.



The guides around your object change depending on what tool you have selected.

Here is what it looks like when you're scaling the object.

Notice if you move the blue, red or green boxes, your object will squish or stretch. To scale uniformly, move the **orange** boxes.

Left click and drag to change your view, right click and drag to move the plate in your view. Scroll wheel to zoom.

Here is another place you can scale your object more precisely. Here you can increase it by percentages...

This button will say **"Slice Now"** while you're still tinkering.

You can click it when you think you're finished and it will update to say **"Export G-code"** as well as show you the estimated print time above.

Object manipulation

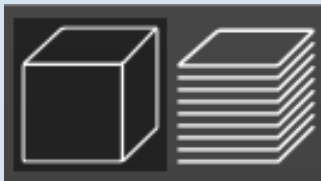
	X	Y	Z	
Position:	4.92	4.13	0.5	in
Rotate:	0	0	0	°
Scale factors:	100	100	100	%
Size:	0.96	0.76	1	in

☒ Inches

...and here you can change the different dimensions in inches.

Click the little lock icon if you want it stretch or squish the object instead of scaling it uniformly.

Slice now

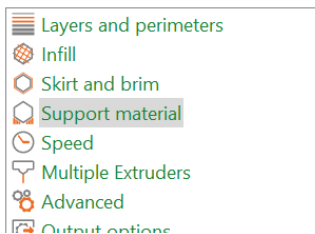


In the lower left, you'll see these two icons. Once you have clicked **"Slice now"** The second one that looks like a stack will be highlighted as this shows you the print preview. To go back and edit some more, just click the solid box icon again. After making changes, you'll just need to click the **"Slice Now"** button again.

For more options, look at the tool box at the top of the screen! We actually don't use the icons much, but from here you can add another object, remove an object, duplicate, and arrange what you have on your build plate.



What we'll actually use, however, is the menu above. Click on "Print Settings"



The screen will change (don't worry, your object is still back in the "Plater" view - you can click back and forth to reassure yourself).

On the left, you'll see this menu appear. There is a lot you can change in here, but we'll focus on "Support Material".

Auto generated supports are usually good enough, but to be sure, you can always click "Generate support material" and choose what degree of overhang gets support!

You should never actually need a printed raft on the Prusa! It auto-levels its build plate before every print.

The contact Z distance will save you a lot of grief when you are removing your support material from the print!

Change this to 0.2 - it leaves a little gap between the supports and the print!

Support material

☒ Generate support material:

☒ Auto generated supports:

☐ Overhang threshold: 55 °

Raft

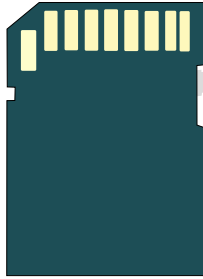
☒ Raft layers: 0 layers

Options for support material and raft

☒ Contact Z distance: 0.2 mm

☒ Pattern: Rectilinear

Now to get it to the printer...



Grab one of our SD cards and insert it into the SD port on the right side of the laptop. Now when you're ready to export your file in PrusaSlicer, the SD card will show up as a destination.

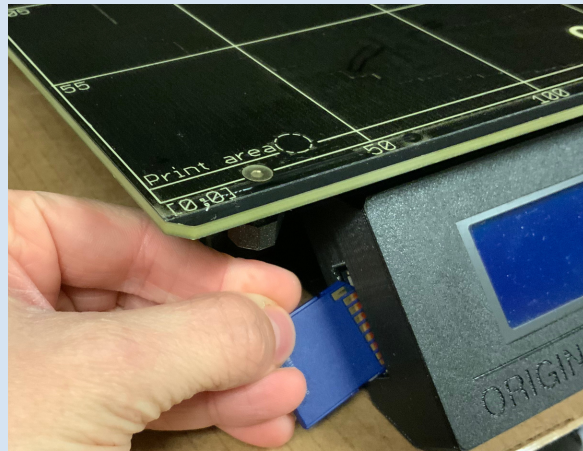
Back in PrusaSlicer, click the button that now says **"Export G-Code"** and select the SD card in the lefthand menu of the window that appears.

Give your file a short, memorable name and save it (it will be saved as a GCODE file).

Now all you have to do is take the SD card and put it into the Prusa!

For whatever reason, the Prusa's SD card port is backwards, so you'll flip the card around so that the gold strips are facing out when you plug it into the port (as shown to the right).

Just push it gently until you feel the spring grab it.



Sometimes stuff goes wrong. 3D printing isn't a perfect science and while you can do everything right, there are sometimes factors that will affect a print job and make it turn into plastic spaghetti. These factors can include, but are not limited to:

- The ambient temperature - if it is too cold in the room, a print might cool too quickly and not stick to the bed.
- A clogged nozzle - if a glob of filament heats unevenly, it can get lodged in the nozzle or come out as more of a blob and affect the print quality.
- A loose belt - a 3D printer uses rubber belts to guide it along its x and y axes, if one slips or gets a little loose, the print can come out crooked.
- Filament gone bad - We keep our filament in humidity controlled cases, but sometimes you can still get dried filament or filament with pockets of moisture snuck in and it can effect the print.

Basically if anything goes wrong associated with any of these, the Emerging Tech Librarian will be there to (try to) help!